National Defence / Défense nationale

AN EVALUATION OF THE HSPICE/RADSPICE CIRCUIT SIMULATION CODE SYSTEM (U)

by

G.T. Pepper and R.E. Stone
Nuclear Effects Section
Electronics Division

DEFENCE RESEARCH ESTABLISHMENT OTTAWA
TECHNICAL NOTE 90-33

PCN
041LS

November 1990
Ottawa

ABSTRACT

The RADSPICE computer code (an optional yet integral component of the HSPICE circuit simulation computer code, copyright META-SOFTWARE Inc.) can be used to simulate radiation effects in electronic circuits. This report has been produced to summarize the capabilities of the RADSPICE code and to increase the amount of detailed documentation available to the RADSPICE user. A general description of RADSPICE (including how to construct electronic circuit network listings or "netlists") has been provided and the range of applications, ease of use (including "manufacturer" support available to users) and accuracy of results obtained with RADSPICE have been discussed. It has been found that effective use of HSPICE/RADSPICE demands much from the user. The user must be familiar with the many assumptions and approximations built into the code system, be able to apply correctly these approximations and assumptions in creating a numerical model of the device(s) to be simulated and finally, be sophisticated enough to interpret accurately the results of the simulation. Given that these concerns are understood and that appropriate input data is supplied to the netlist for the intended simulation, HSPICE/RADSPICE can be used to simulate both electronic circuits and radiation effects in electronic circuits.

RÉSUMÉ

Le code machine RADSPICE est une option dans le code machine HSPICE qui peut être utilisé pour simuler les effets de la radiation dans des circuits électroniques. Ce rapport a été produit dans le but de faire un résumé sur le potentiel du code RADSPICE et d'augmenter l'information disponible à l'utilisateur. Il nous donne également une description générale du code RADSPICE en incluant la façon de construire une liste ou "netlists" du réseau électronique. En plus, nous discutons du champ d'application, de la facilité d'utilisation (incluant l'aide du "manufacturier" qui est disponible aux usagers) et de l'exactitude des résultats obtenus avec RADSPICE. L'utilisateur doit démontrer beaucoup d'ingéniosité pour se familiariser avec le code HSPICE/RADSPICE. L'utilisateur doit également être familier avec toutes les hypothèses et approximations bâtis dans le système, appliquer correctement ces hypothèses et ces approximations en créant un model numérique du dispositif à être simulé et finalement, être capable d'interpréter les résultats de la simulation. En prenant pour acquis que le tout a été compris et que les bonnes données ont été fournies à la simulation, HSPICE/RADSPICE peut être utilisé pour simuler les circuits électroniques et les effets de la radiation sur les circuits électroniques.

EXECUTIVE SUMMARY

The study of radiation effects in modern electronic devices is currently of military and commercial interest. Detailed research in this area necessarily requires either extensive experimentation involving actual radiation sources and semiconductor devices (extremely resource-intensive) or accurate numerical simulations to substitute for the aforementioned experiments (a much more cost-effective method). Since the HSPICE/RADSPICE code system permits detailed numerical simulations of several types of radiation effects in modern electronic devices and circuits, it has been selected for DREO use in preliminary investigations in this field.

This report has been produced to summarize the capabilities of the RADSPICE code and to increase the amount of documentation available to the RADSPICE user. A general description of RADSPICE has been provided, including the range of applications and ease of use (i.e. how to construct electronic circuit network listings or "netlists", "manufacturer" support available to users) of the code. Commentary upon the accuracy of results obtained with RADSPICE has been provided. Although an experienced user will find HSPICE/RADSPICE relatively flexible in its range of applications, all users must be strongly cautioned that HSPICE/RADSPICE is not a "black-box" which can be applied indiscriminately in simulating any and all types of radiation effects in electronic circuits.

TABLE OF CONTENTS

		PAGE
	ABSTRACT/RÉSUMÉ	iii
	EXECUTIVE SUMMARY	iv
	TABLE OF CONTENTS	v
	LIST OF FIGURES	vi
1.0	INTRODUCTION	1
2.0	GENERAL DESCRIPTION OF RADSPICE	1
2.1	EFFECTS IN DEVICES	1
2.2	MODELLING METHODS	2
2.2.1	INVOKING TOTAL DOSE EFFECTS IN RADSPICE	3
2.2.2	INVOKING NEUTRON FLUENCE EFFECTS IN RADSPICE	3
2.2.3	INVOKING PHOTOCURRENT EFFECTS IN RADSPICE	3
3.0	USING HSPICE/RADSPICE	6
3.1	INPUT REQUIREMENTS	6
3.2	OUTPUT CAPABILITY	11
3.3	EASE OF USE OF HSPICE/RADSPICE	17
3.4	ACCURACY OF RESULTS OBTAINED WITH HSPICE/RADSPICE	18
4.0	CONCLUSIONS	18
5.0	REFERENCES	20

LIST OF FIGURES

PAGE

FIGURE 1 : RADSPICE simple diode circuit for simulation of photocurrent produced by diode exposure to a square pulse of ionizing radiation 10^6 rad/s dose-rate (input netlist - "TESTDI.SP") 8

FIGURE 2 : Photocurrent and ionizing radiation pulse profile (dose-rate versus time) for simple diode circuit of "TESTDI.SP" input netlist/"TESTDI.LIS" output file. . 15

FIGURE 3 : HSPLOT version of photocurrent for simple diode circuit of "TESTDI.SP" input netlist/ "TESTDI.LIS" output file 16

1.0 INTRODUCTION

The HSPICE/RADSPICE computer code system has been jointly developed by META-SOFTWARE Inc. and Science Applications International Corporation (SAIC). The code system can be used to simulate the design, modification and optimization of modern radiation-hardened electronic circuits and components. As a result of this simulation, design costs can be minimized through a reduction in the number of iterations required for prototype fabrication and refinement.

Specifically, HSPICE is an optimizing analog circuit simulator capable of typical SPICE (Simulation Program with Integrated Circuit Emphasis) analyses (i.e. A.C. or D.C., steady-state, transient and frequency domain analyses), while the RADSPICE (optional subprogram) permits simulation of several types of radiation effects in modern electronic devices. Since the study of radiation effects in modern electronic devices is currently of military and commercial interest, RADSPICE has been selected for preliminary investigations in this field.

This report has been produced to summarize the capabilities of the RADSPICE computer code and to increase the amount of detailed documentation available to the RADSPICE user. Specifically, this report presents a general description of RADSPICE, including the range of applications and ease of use (i.e. how to prepare netlists, "manufacturer" support available to users) of the code. Commentary upon the accuracy of results obtained with RADSPICE has been provided. However, since RADSPICE is an extension of HSPICE, areas exist where both parts of the code system must be discussed concurrently.

2.0 GENERAL DESCRIPTION OF RADSPICE

2.1 EFFECTS IN DEVICES

In general, active semiconductor components (metal-oxide-semiconductor field- effect transistors or "MOSFETs", metal-semiconductor field-effect transistors or "MESFETs", junction field-effect transistors or "JFETs", bipolar junction transistors or "BJTs", junction diodes) of electronic circuits are relatively vulnerable to the effects of ionizing radiation, while passive components (resistors, capacitors, inductors) are far less sensitive. Hence, radiation "hardening" efforts for electronic circuits should necessarily be directed toward active component selection, design, fabrication and application. Therefore, RADSPICE has been developed to simulate radiation-induced effects only in active semiconductor components of electronic circuits.

RADSPICE can be used to model three main radiation effects in several types of semiconductor components; permanent degradations in MOSFETs caused by total dose effects, permanent degradations in BJTs caused by exposure to neutron fluence, and finally, transient photocurrent effects in p-n junctions of diodes, BJTs, JFETs, MOSFETs and MESFETs. Effect selection is performed through the RADSPICE ".OPTIONS" statement. Total dose effects in a specific device are modelled by providing the level of the total dose. Transient photocurrent effects are modelled by specifying the pulse-shape and the dose-rate (rad/s) supplied to the device. Additionally, RADSPICE provides the graphics display package "HSPLOT" to display simulation results, along with a standardized active electrical "device data library" or DDL (available only as precompiled macro models) to facilitate simulation of circuits containing "industry-standard" components.

2.2 MODELLING METHODS

RADSPICE supports modelling of total dose and dose-rate effects in Si, Ge and GaAs semiconductor devices. Device composition selection can be made by choosing the appropriate "carrier generation constant" (4.2×10^{19} electron-hole pairs or ehp/(m^3-rad) in Si, 1.1×10^{20} ehp/(m^3-rad) in Ge and/or 6.63×10^{19} ehp/(m^3-rad) in GaAs) for the respective type of device material. Additionally, other material-dependent physical input parameters (such as band-gap voltage) must be defined to reflect the choice of carrier generation constant.

HSPICE/RADSPICE determines the electrical characteristics of devices undergoing simulation through user initialization of the "LEVEL" parameter in the ".MODEL" section of the netlist. The "LEVELs" of models are based upon different theoretical or empirical electrical descriptions of a device, and vary in degree of complexity from level to level. Proprietary "LEVELs" have been developed to describe particular vendors' parts produced by their own particular (proprietary) manufacturing processes. Due to the proprietary nature of these part (device) models, RADSPICE source code is not included with the code package supplied by META-SOFTWARE. New electrical models (prompted by expansion of product lines of private clients such as Siemens and AMD) are constantly under development, with corresponding new "LEVELs" added to HSPICE/RADSPICE shortly thereafter.

A MOSFET example can be used to illustrate the extensive set of electrical levels of device models available for accurate simulation of device electrical characteristics, with 35 electrical variants or levels available ("n" ranges from 1 through 35) to the user. A specific example can be seen for the MOSFET LEVEL=27 case, where placement of the "LEVEL=27" statement in the ".MODEL" section of a netlist informs HSPICE/RADSPICE that a SOSFET (Silicon-on-sapphire FET) device is to be included in the simulation.

Once the electrical characteristics of the device are defined, the user can then select the most appropriate of several methods available for simulation of various radiation effects in semiconductor devices. The following subsections provide descriptions of these methods.

2.2.1 INVOKING TOTAL DOSE EFFECTS IN RADSPICE

RADSPICE invokes the total dose radiation effect by including the ".OPTIONS DOSE=value" statement in the netlist of the simulated component or circuit. The "value" portion of the command string represents the level of dose (in rads) to be deposited in the device undergoing simulation. Actual mathematical degradation of MOSFET parameters is performed by applying linear degradation equations to the MOSFET threshold voltage, low field mobility and the fast surface state density. RADSPICE requires that [1] three MOSFET model parameters - "DVT", "DMU" and "DNFS" - are included in the ".MODEL" section of the netlist as proportionality factors for the above three effects (respectively).

Caution must be exercised when evaluating MOSFET total dose simulation results, due to the first-order nature of the degradation equations. Ideally, any set of model parameters should be based on actual measurements made across the entire range of total doses received by the MOSFET. If such empirically-derived parameters are available, simulation results would generally be more accurate than obtained from simulations conducted in the absence of an empirical database. Presently, total dose effects are available only for MOSFET device simulation. However, RADSPICE documentation states that total dose simulation in other types of semiconductor devices will become available at a later date.

2.2.2 INVOKING NEUTRON FLUENCE EFFECTS IN RADSPICE

RADSPICE does not support modelling of neutron fluence effects in active devices at the present time. However, current RADSPICE documentation states that neutron-induced gain degradation effects in BJTs will become available at a later date.

2.2.3 INVOKING PHOTOCURRENT EFFECTS IN RADSPICE

RADSPICE photocurrent models for diodes, BJTs, JFETs and MOSFETs can be invoked by selection of the ".OPTIONS RAD=n" options statement. The "n" term in the options statement is provided for user control of the type of photocurrent analysis required, as discussed below;

(1) if "RAD=0", photocurrent modelling is not done,

(2) if "RAD=1", photocurrents are calculated for all p-n junctions in all devices specified in the netlist of the problem. Note that photocurrent simulation is performed for the base-collector junction of BJTs only, since this junction is the primary contributor to BJT photocurrent production and

(3) if "RAD=2", photocurrent calculation is performed as per (2) above, however, the emitter-base junction of BJTs is also assumed to generate photocurrents.

A photocurrent source function must be supplied in the RADSPICE netlist to define the dose-rate time-history and type of photocurrent model equation(s) used in the simulation. The general photocurrent source function is defined;

"Zm1 PHOTO RLEV=m2 function definition".

In this example, "m1" represents the order of appearance of the photocurrent source in the netlist (m1=1 would imply that this would be the first source defined in the netlist). The "m2" variable represents the photocurrent model equation to be used by all p-n junctions using this model. The "function definition" substatement represents any of the RADSPICE-defined independent source functions (PULSE, SIN, PWLF, EXP, etc.) available to the user for representing the time history of the radiation.

RADSPICE uses the previously-defined photocurrent sources to determine the time behaviour of the photocurrents in each junction. Up to nine different photocurrent sources, "m1" from above, can be included in a single circuit simulation. Photocurrents can be calculated using different models, "m2" from above, with each model defined by different parametric equations. The radiation effects model parameters are necessarily different for different RLEV ("m2") values, since different radiation effects models are used. Note that "RLEV" selection is not conditional upon initialization of the "LEVEL" parameter. "LEVEL=n" is used to describe the electrical characteristics of the type or model of semiconductor device undergoing simulation (Siemens MOSFET, AMD BJT, etc.), while "RLEV" is used to select the type of radiation effects model to be employed in the RADSPICE simulation for the device.

If RLEV=1 is selected, "function definition" must be chosen as "PULSE" (as RLEV=1 defines the Wirth-Rogers analytic solution for junction photocurrents which is valid only for a

square pulse of given dose-rate). Since RLEV=2 has not been implemented by META-SOFTWARE, it is reserved for further use. However, RLEV=3 is chosen to indicate that an arbitrary user-supplied (excitation source) photocurrent is to be placed in parallel with each referenced junction and the response of the referenced junction (or of the complete circuit) is to be observed. RADSPICE arbitrary photocurrent sources are presented below. Other arbitrary photocurrent functions are presented in section 3 of Ref. 1.

(1) Pulse - square pulse of photocurrent. Input requirements include pulse period, width, rise and fall times, initial and final magnitudes of pulse (i.e. dose-rate).

(2) Sinusoidal - requires frequency, phase delay (in degrees and in seconds), damping factor (for decaying or increasing sinusoids), initial and final magnitudes of waveform (i.e. dose-rate).

(3) Exponential - requires rise and fall time constants, rise and fall delay times, initial and final magnitudes of waveform (i.e. dose-rate).

(4) Piecewise Linear Fit - consecutively appended line segments of photocurrent. Requires delay time as well as time and magnitude datapoints (i.e. dose-rate) for each of the segments of the waveform.

The above photocurrent sources can be configured as dose-rate-dependent sources if necessary. Presently, the RLEV=3 model option could be of particular interest to DREO researchers as it can be used to model transient photocurrents that initiate single-event upset (SEU) in semiconductor devices. Should it be desirable to simulate SEU with RADSPICE (presently not an explicit option), the analyst can supply one of the arbitrary current sources to the device junction under study to simulate the passage of a heavy ion in the semiconductor material. Such simulations have been performed for a 1 Mb static RAM (SRAM) cell - both by DREO researchers and by researchers at the United States Naval Research Labs [4] - with the results of DREO simulations presented in a current DREO report [3].

Finally, for RLEV=4, 5 or 6, photocurrent modelling equations are invoked which are close approximations to the junction photocurrents (unlike the Wirth-Rogers approach [2] where an exact solution is produced for a fixed-potential junction receiving a square dose-rate pulse). Selection of these options provides the user with arbitrary junction voltages and dose-rate time responses from which a device-specific photocurrent model can be constructed. For a complete description of the various options, models and associated parameters, see pp. 17-9 through 17-16 of Ref. 1.

3.0 USING HSPICE/RADSPICE

The HSPICE/RADSPICE code system is available from META-SOFTWARE for every major computer platform from the 80386 PC to the CRAY supercomputer. Software is supplied by META-SOFTWARE as executable files in binary format, written and compiled for the computer platform specified by the client. Note that META-SOFTWARE restricts RADSPICE access to licensed users only by requiring that a coded hardware security key is installed on the parallel printer port (LPTI) of the authorized user's 80386 PC prior to program execution. "HSPLOT" (a primitive graphics program) is also included with the HSPICE/RADSPICE code system. HSPLOT is menu-driven and is capable of generating output hardcopy on a variety of devices including plotters, dot matrix printers and laser printers. Selected output parameters are included in an HSPLOT intermediate plot file, ("FILENM.TR0" for TRansient plots, "FILENM.SW0" for D.C. SWeep plots), with this intermediate file processed by HSPLOT and supplied to one of the aforementioned output devices for hardcopy.

Typical HSPICE/RADSPICE simulations may require from seconds to hours for completion, depending upon the complexity of the simulation and host computer platform.

3.1 INPUT REQUIREMENTS

HSPICE/RADSPICE input data required to construct a simulation can include a description of

(1) circuit topology and required electronic components, arranged in a circuit representative "netlist" (HSPICE and RADSPICE),

(2) individual device composition (Si, GaAs or Ge), geometry and electrical parameters required for the analysis (HSPICE and RADSPICE),

(3) the type of analysis required, i.e. transient, steady-state or frequency domain (HSPICE and RADSPICE),

(4) the radiation effects parameters required to simulate the incident radiation pulse for either fixed pulse(s) or user-defined arbitrary pulse(s) (RADSPICE ONLY),

(5) the radiation effects parameters required to define the device response to the incoming radiation pulse (RADSPICE ONLY) and

(6) initialization of the options required to select output parameters of interest, to define the simulation output format and to create an HSPLOT input plot file (HSPICE and RADSPICE).

Once this information is made available to HSPICE/RADSPICE via a (general) input netlist, "FILENM.SP", an HSPICE/RADSPICE simulation can then be invoked simply by typing HSPICE FILENM on the user's computer terminal keyboard. A comprehensive list of all HSPICE/RADSPICE input options is available in Ref. 1, pp. 2-1 through 2-42.

The "TESTDI.SP" RADSPICE sample netlist can be found below. This netlist describes a square pulse of ionizing radiation (x-rays, gamma rays, and/or electrons) yielding a dose-rate of 10^8 rad/s, incident upon the diode-resistor circuit of Fig. 1.

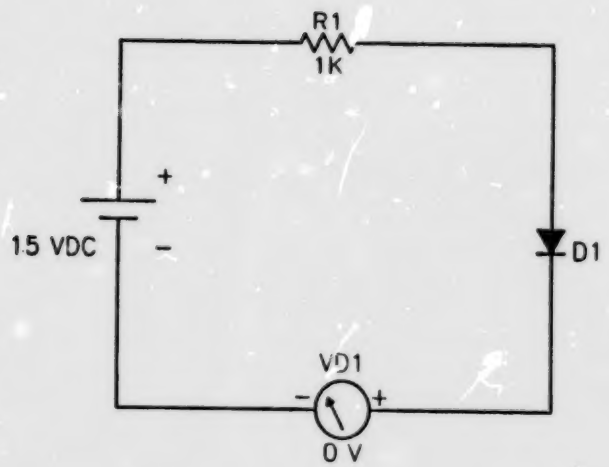

FIGURE 1 : RADSPICE simple diode circuit for simulation of photocurrent produced by diode exposure to a square pulse of ionizing radiation of 10^6 rad/s dose-rate (input netlist - "TESTDI.SP").

```
*****************************************************************************
*
* FILE : TESTD1.SP - PHOTOCURRENT IN STANDARD P-N DIODE.  TOPOLOGY
*        CONSISTS OF SIMPLE DIODE, 1K RESISTOR AND 15 VDC POWER
*        SUPPLY.
*
* RLEV=3 EXAMPLE - ARBITRARY USER-DEFINED SQUARE PULSE OF DOSE-RATE.
*
*****************************************************************************
*
* DESCRIBE THE RADIATION EFFECTS CONTROL PARAMETERS REQUIRED FOR
* THE SIMULATION...
*
.OPTIONS RAD=1
*
* INCLUDE HSPLOT OUTPUT...
*
.OPTIONS POST=1
*
*****************************************************************************
*
* SELECT I/O CONTROL AND CALCULATION PARAMETERS FOR THE SIMULATION...
*
.CONTROL ACCT INGOLD=2 DCSTEP=0 ABSI=1.0E-17 NOTOP
+ NUMDGT=7 PIVTOL=1.0E-30 GMIN=1.0E-25 RELI=1.0E-3
*
*****************************************************************************
*
* DESCRIBE THE CIRCUIT TOPOLOGY...
*
VCC 7  0  DC 15
R1  2  1  1K
VD1 7  2  DC  0
D1  0  1  DIO1
*
*****************************************************************************
*
* DESCRIBE THE DOSE-RATE SUPPLIED TO THE DIODE...
*
Z1 PHOTO RLEV=3 PULSE (0 1e6 5u .1u .1u 20u 50u)
*
*****************************************************************************
*
* SELECT TYPE OF ANALYSIS AND THEN PLOT DIODE PHOTOCURRENT...
*
.TRAN 1u 50u
.PRINT TRAN I1(VD1)
.PLOT TRAN I1(VD1)
*
*****************************************************************************
*
* DESCRIBE THE ELECTRONIC PARAMETERS OF THE DIODE UNDER SIMULATION...
*
.MODEL DIO1 D(
+ LEVEL=1
+ IS=  5.5E-7   CJO=  2.5P   TT  = 2.1N    RS  =  .50
+ PB=   .8        M=   .5    DIFL= 2.0E-6  AJ  = 2.25E-8
+ GEN= 4.0E19    TD= 90N     SRC = 1       RFAC= 1.      )
*
*****************************************************************************
*
* END THE SIMULATION...
*
.END
```

Some of the radiation effects parameters included in the "TESTDI.SP" netlist are;

(1) RAD - used to enable photocurrent calculation in a p-n junction,

(2) RLEV - used to select Wirth-Rogers analytic solution to junction photocurrents induced by a square pulse of ionizing radiation,

(3) 1e6 - ionizing radiation dose-rate (rad/s) supplied to diode,

(4) 5u - time that arrival of ionizing radiation pulse is delayed after simulation is initiated (u = microseconds),

(5) .1u - respectively, the rise and fall times of the ionizing radiation pulse (coincidentally equal in this simulation),

(6) 20u - duration of pulse,

(7) 50u - duration of simulation,

(8) RFAC - user-defined dimensionless dose-rate or photocurrent multiplication factor,

(9) GEN - carrier generation constant for (Si) diode material

and

(10) SRC - radiation dose-rate source selector.

Other "TESTDI.SP" netlist parameters are described in Ref. 1.

3.2 OUTPUT CAPABILITY

HSPICE/RADSPICE simulation output data can be tailored to suit several user-specified formats. Typical HSPICE/RADSPICE simulation output (generated by the "TESTDI.SP" netlist) includes;

(1) a comprehensive list of all input (user-specified and default) parameters used in the analysis,

(2) a comprehensive list of all user-specified calculated parameters,

(3) the response of the circuit (i.e. transient, steady-state or frequency domain) at user-specified nodes throughout the circuit. Also, circuit response in or at a specific circuit component node can be examined with HSPICE/RADSPICE and

(4) plot files for the HSPLOT routine, used to graphically display optioned output data.

A comprehensive list of all HSPICE/RADSPICE output control options is available in Ref. 1, pp. 13-1 through 13-30. The "TESTDI.LIS" sample output created by the execution of the "TESTDI.SP" netlist appears below. The photocurrent produced in the diode by the radiation pulse is available in tabular form at the end of the "TESTDI.LIS" output file and appears graphically in Fig. 2. The ionizing radiation pulse also appears in Fig. 2. Figure 3 shows the diode photocurrent of the the "TESTDI.*" simulation, in typical HSPLOT form.

```
**********************************************************************
TESTDI.LIS :

******    h s p i c e     9001a          10:23:29  09 nov90   pc
**********************************************************************
******  input listing                   maintainance renewal 910630
******
*
* file : testdi.sp - photocurrent in standard p-n diode.  topology
*        consists of simple diode, 1k resistor and 15 vdc power
*        supply.
*
* rlev=3 example - arbitrary user-defined square pulse of dose-rate.
*
**********************************************************************
*
* describe the radiation effects control parameters required for
* the simulation...
*
.options RAD=1
*
* include hsplot output...
*
.options post=1
*
```

```
**********************************************************************
*
* select i/o control and calculation parameters for the simulation...
*
.control acct ingold=2 dcstep=0 absi=1.0e-17 notop
+ numdgt=7 pivtol=1.0e-30 gmin=1.0e-25 reli=1.0e-3
*
**********************************************************************
*
* describe the circuit topology...
*
vcc 7  0  dc 15
r1  2  1  1k
vd1 7  2  dc  0
d1  0  1  dio1
*
**********************************************************************
*
* describe the dose-rate supplied to the diode...
*
z1 photo rlev=3 pulse (0 1e6 5u .1u .1u 20u 50u)
*
**********************************************************************
*
* select type of analysis and then plot diode photocurrent...
*
.tran 1u 50u
.print tran i1(vd1)
.plot tran i1(vd1)
*
**********************************************************************
*
* describe the electronic parameters of the diode under simulation...
*
.model dio1 d(
+ level=1
+ is=  5.5e-7   cjo=  2.5p   tt  = 2.1n    rs  =  .50
+ pb=   .8        m=   .5    difl= 2.0e-6  aj  = 2.25e-8
+ gen= 4.0e19    td= 90n     src = 1       rfac= 1.      )
*
**********************************************************************
*
* end the simulation...
*
.end
*
*
*
******    hspice +rad      9001a          10:23:29  09 nov90   pc
**********************************************************************
******  diode model parameters          tnom=  25.000 temp=  25.000
******
************************************************************
*** common model parameters model name:   0:dio1    ****
************************************************************

    names  values   units     names  values   units     names  values   units
    -----  ------   -----     -----  ------   -----     -----  ------   -----

  1*** dc breakdown parameters ***
     vb=   0.     volts        ibv=   1.00m amps

  2*** parasitic resistance parameters ***
     rs= 500.00m ohms

  3*** capacitance parameters ***
     fc= 500.00m               fcs= 500.00m                 m= 500.00m
   mjsw= 330.00m                pb= 800.00m volts         php= 800.00m volts
     tt=   2.10n secs          cjo=   2.50p f/area        cjp=   0.    f/pj
```

```
 4*** temperature effect parameters ***
   tlev=   0.                tlevc=   0.                 tcv=   0.     v/deg k
    trs=   0.     /deg         cta=   0.     /deg         ctp=   0.     /deg
    xti=   3.00                eg=   1.11   ev          gap1= 702.00u   ev/deg
   gap2=   1.11k  deg         tt1=   0.     /deg         tt2=   0.     /deg2
    tm1=   0.     /deg         tm2=   0.     /deg2        tpb=   0.     v/deg k
   tphp=   0.     v/deg k     tref=  25.00   deg c

 5*** noise parameters ***
     kf=   0.                  af=   1.00

 6*** radiation parameters ***
    src=   1.00               rfac=   1.00                 aj=  22.50n m**2
   difl=   2.00u meters       gens=   0.     car/RAD       tds=   0.     secs
   fdep=   0.                   td=  90.00n secs          gen= 4.0e+19 note1

    ### note1  represents  car/RAD/m**3  ###

 ***  level   1  model parameters  ***

  level=   1.00                 is= 550.00n amps          jsw=   0.     amps
      n=   1.00                 ik=   0.     amp          area=   1.00
     pj=   0.                  ikr=   0.     amp
*
*
*
******    hspice +rad      9001a          10:23:29  09 nov90   pc
*******************************************************************************
******  operating point information      tnom=  25.000 temp=  25.000
******
***** operating point status is voltage   simulation time is     0.
   node    =voltage         node    =voltage         node    =voltage


+0:1       = 1.499945e+01 0:2       = 1.500000e+01 0:7       = 1.500000e+01
*
*
*
******    hspice +rad      9001a          10:23:29  09 nov90   pc
*******************************************************************************
******  transient analysis               tnom=  25.000 temp=  25.000
******

                diode
    time        current
                 vd1
  0.           5.500150e-07
 1.0000000e-06 5.500000e-07
 2.0000000e-06 5.500000e-07
 3.0000000e-06 5.500000e-07
 4.0000000e-06 5.500000e-07
 5.0000000e-06 5.500000e-07
 6.0000000e-06 1.571739e-02
 7.0000000e-06 1.571739e-02
 8.0000000e-06 1.571739e-02
 9.0000000e-06 1.571739e-02
 1.0000000e-05 1.571739e-02
 1.1000000e-05 1.571739e-02
 1.2000000e-05 1.571739e-02
 1.3000000e-05 1.571739e-02
 1.4000000e-05 1.571739e-02
 1.5000000e-05 1.571739e-02
 1.6000000e-05 1.571739e-02
 1.7000000e-05 1.571739e-02
 1.8000000e-05 1.571739e-02
 1.9000000e-05 1.571739e-02
 2.0000000e-05 1.571739e-02
 2.1000000e-05 1.571739e-02
 2.2000000e-05 1.571739e-02
```

```
 2.3000000e-05  1.571739e-02
 2.4000000e-05  1.571739e-02
 2.5000000e-05  1.571739e-02
 2.6000000e-05  2.677708e-06
 2.7000000e-05 -1.346047e-06
 2.8000000e-05  2.441351e-06
 2.9000000e-05 -1.337151e-06
 3.0000000e-05  2.432914e-06
 3.1000000e-05 -1.328674e-06
 3.2000000e-05  2.424456e-06
 3.3000000e-05 -1.320235e-06
 3.4000000e-05  2.416036e-06
 3.5000000e-05 -1.311835e-06
 3.6000000e-05  2.407654e-06
 3.7000000e-05 -1.303471e-06
 3.8000000e-05  2.399309e-06
 3.9000000e-05 -1.295146e-06
 4.0000000e-05  2.391003e-06
 4.1000000e-05 -1.286858e-06
 4.2000000e-05  2.382733e-06
 4.3000000e-05 -1.278607e-06
 4.4000000e-05  2.374500e-06
 4.5000000e-05 -1.270393e-06
 4.6000000e-05  2.366305e-06
 4.7000000e-05 -1.262216e-06
 4.8000000e-05  2.358146e-06
 4.9000000e-05 -1.254075e-06
 5.0000000e-05 -2.300480e-06
        ***** job concluded
*
*
*
******    hspice +rad      9001a          10:23:29  09 nov90   pc
**********************************************************************
******  job statistics summary           tnom=  25.000 temp=  25.000
******

 # nodes =      6 # elements=      6 # real*8 mem avail/used=  602000/    3994
 # diodes=      1 # bjts    =      0 # jfets   =      0 # mosfets =      0

    analysis      time      # points  tot. iter  conv.iter

    op point          .28         1        10
    transient        6.20        51       391       127 rev=    6
    input+setup      1.76
    pass1             .77
    readin            .77
    errchk            .11
    setup             .11
    output            .49

            total cpu time         10.05 seconds
            job started at  10:23:29  09 nov90
            job ended   at  10:23:44  09 nov90
```

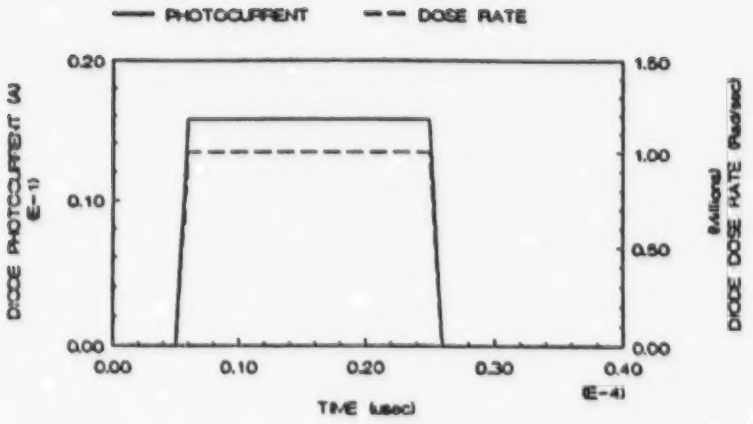

FIGURE 2 : Photocurrent and ionizing radiation pulse profile (dose-rate versus time) for simple diode circuit of "TESTDI.SP" input netlist/ "TESTDI.LIS" output file.

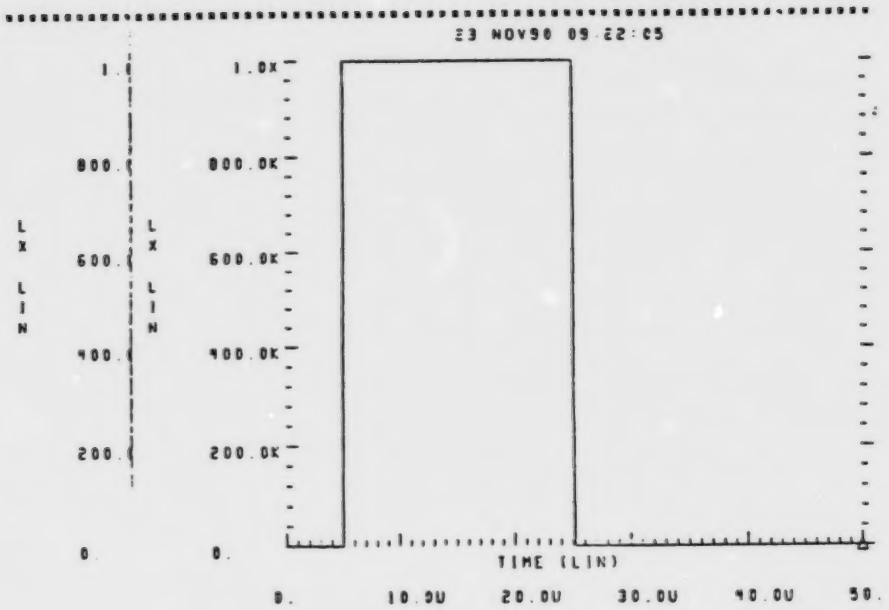

FIGURE 3 : HSPLOT version of photocurrent for the simple diode circuit of "TESTDI.SP" input netlist/"TESTDI.LIS" output file.

3.3 EASE OF USE OF HSPICE/RADSPICE

The previous HSPICE/RADSPICE simple diode model demonstrates that complicated command procedures, terms, symbols and statements are unnecessary for code system production of sensible output data. All required terms and symbols are generally well-defined [1], with META-SOFTWARE supplying user support through a toll-free "hotline".

An extensive directory of sample problems - complete with input and output files for diodes, BJTs, JFETS and MOSFETs - is included with the code package. Also, the DDL (available as a no-cost option from META-SOFTWARE) contains a relatively complete list of common semiconductor devices. These devices may be included in HSPICE/RADSPICE netlists via the ".MODEL" statement for any netlist. However, the user must be cautioned that the DDL contains "typical" parameterized electrical data only (e.g. gain, I-V curves). Physical data of interest to the radiation effects analyst (i.e. junction area and width, diffusion length) are not included in the dataset due to the proprietary nature of these physical parameters. Also, manufacturers are reluctant to release physical parameters, simply because for a given device (i.e. part number) these parameters can vary widely - not only from manufacturer to manufacturer but from part lot to part lot.

Additionally, several time-and-labour-saving options have been included in HSPICE/RADSPICE that permit completion of multiple runs with one netlist (valuable at the "scoping" or "optimizing" stage of component or circuit analysis). Interestingly, META-SOFTWARE states that the sophisticated user can construct a RADSPICE netlist that will invoke "auto-optimization" analyses for devices undergoing simulation (selected output parameters are calculated, compared to defined standards and optimized if necessary). Also, the auto-optimization feature includes an n-dimensional regression analysis that can be used to extract parameters of interest from measured device data (including parameters related to the response of devices to ionizing radiation). RADSPICE users interested in this capability are advised to contact META-SOFTWARE for more information, since this is a newly-implemented, undocumented feature.

Finally, although the code system is relatively easy to use, all users must be aware that HSPICE/RADSPICE is not a "black-box" that can be applied indiscriminately for simulating any and all types of electronic circuits. The user must be familiar with the many assumptions and approximations built into the code system. Also,

the user must be sophisticated enough to be able to apply the approximations and assumptions present in the code system to the numerical description of the device(s) for which the simulation is intended. Given that these concerns are understood and that appropriate input data is supplied to the netlist for the intended simulation, HSPICE/RADSPICE can be used to simulate both electronic circuits and radiation effects in electronic circuits.

3.4 ACCURACY OF RESULTS OBTAINED WITH HSPICE/RADSPICE

At the time of this report, HSPICE/RADSPICE has been undergoing evaluation at DREO for approximately six months. During this evaluation period RADSPICE has been widely used in simulating various radiation effects in numerous semiconductor devices at the discrete and integrated circuit (IC) level. The accuracy of results obtained from semiconductor device simulations performed with RADSPICE depends primarily upon the sophistication of the user and the quality of the netlist physical device data at the user's disposal. A comprehensive presentation of the results of the simulations and a commentary upon the accuracy of the associated results are available in Ref. 3.

4.0 CONCLUSIONS

An assessment of the ability of RADSPICE (an HSPICE optional subprogram) to model radiation effects in semiconductor electronic devices has been performed. A general description of RADSPICE (including how to construct input netlists and "manufacturer" support available to users) has been provided and the range of applications, ease of use and accuracy of results obtained with RADSPICE have been addressed. Although the HSPICE/RADSPICE code system is relatively flexible in its range of applications, it is not a "black-box" to be applied indiscriminately in simulating any and all types of radiation effects in electronic circuits.

The HSPICE/RADSPICE user must be familiar with the many assumptions and approximations inherent in the code system. Then, these approximations and assumptions must be correctly applied during simulation set-up and initiation. Finally, the user must be able to accurately interpret the results of HSPICE/RADSPICE simulations. Assuming that these criteria are met and that appropriate input netlists can be established, HSPICE/RADSPICE will accurately simulate both electronic circuit behaviour and radiation effects in electronics.

5.0 REFERENCES

1. META-SOFTWARE Inc., HSPICE User's Manual H9001, META -SOFTWARE, 1300 White Oaks Rd., Campbell Ca., 95008, U.S.A., 1990.

2. J.L. Wirth and S.C. Rogers, "The Transient Response of Transistors and Diodes to Ionizing radiation", IEEE Transactions on Nuclear Science, Vol. NS-11, no. 5, Nov. 1964, pp. 24-38.

3. G.T. Pepper and R.E. Stone, "Simulation of Radiation Effects in Semiconductor Devices with RADSPICE", DREO Report (to be published), Defense Research Establishment Ottawa, Nov. 1990.

4. A.B. Campbell, "Single-Event Upset Phenomona in Semiconductor Electronics", Seminar presented at Defense Research Establishment Ottawa, Oct. 12, 1990.

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)

DEFENCE RESEARCH ESTABLISHMENT OTTAWA
OTTAWA, ONTARIO
K1A 0Z4

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable)

UNCLASSIFIED

3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title.)

AN EVALUATION OF THE HSPICE/RADSPICE CIRCUIT SIMULATION CODE SYSTEM (U)

4. AUTHORS (Last name, first name, middle initial)

PEPPER, G.T. STONE, R.E.

5. DATE OF PUBLICATION (month and year of publication of document)

DECEMBER 1990

6a. NO. OF PAGES (total containing information. Include Annexes, Appendices, etc.)

24

6b. NO. OF REFS (total cited in document)

4

7. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered.)

DREO TECHNICAL NOTE

8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include the address.)

DEFENCE RESEARCH ESTABLISHMENT OTTAWA
OTTAWA, ONTARIO K1A 0Z4

9a. PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant)

041LS

9b. CONTRACT NO. (if appropriate, the applicable number under which the document was written)

10a. ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)

DREO TECHNICAL NOTE 90-33

10b. OTHER DOCUMENT NOS. (Any other numbers which may be assigned this document either by the originator or by the sponsor)

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(XX) Unlimited distribution
( ) Distribution limited to defence departments and defence contractors; further distribution only as approved
( ) Distribution limited to defence departments and Canadian defence contractors; further distribution only as approved
( ) Distribution limited to government departments and agencies; further distribution only as approved
( ) Distribution limited to defence departments; further distribution only as approved
( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in 11) is possible, a wider announcement audience may be selected.)

NO LIMITATIONS

UNCLASSIFIED

SECURITY CLASSIFICATION OF FORM

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

(U) The RADSPICE computer code (an optional yet integral component of the HSPICE circuit simulation computer code, copyright META-SOFTWARE Inc.) can be used to simulate radiation effects in electronic circuits. This report has been produced to summarize the capabilities of the RADSPICE code and to increase the amount of detailed documentation available to the RADSPICE user. A general description of RADSPICE (including how to construct electronic circuit network listings or "netlists") has been provided and the range of applications, ease of use (including "manufacturer" support available to users) and accuracy of results obtained with RADSPICE have been discussed. It has been found that effective use of HSPICE/RADSPICE demands much from the user. The user must be familiar with the many assumptions and approximations built into the code system, be able to correctly apply these approximations and assumptions in creating a numerical model of the device(s) to be simulated and finally, be sophisticated enough to accurately interpret the results of the simulation. Given that these concerns are understood and that appropriate input data is supplied to the netlist for the intended simulation, HSPICE/RADSPICE can be used to simulate both electronic circuits and radiation effects in electronic circuits.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus, e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it is not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title.)

HSPICE
RADSPICE
SPICE
Radiation Effects
Semiconductor
Netlist
Simulation
Photocurrent
Dose-Rate
Transient Radiation Effects in Electronics

UNCLASSIFIED

SECURITY CLASSIFICATION OF FORM